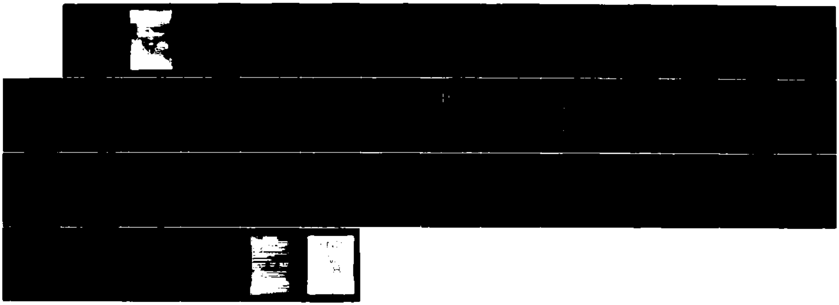
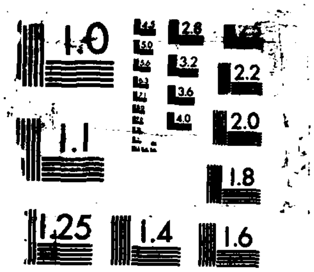
AD-A188 078 A USER'S GUIDE FOR BIE3D - A BOUNDARY INTEGRAL EQUATION 1/1
COMPUTER PROGRAM F..(U) ROYAL AIRCRAFT ESTABLISHMENT
FARNBOROUGH (ENGLAND) K W MAN JUL 86
UNCLASSIFIED RAE-TM-MAT/STR-1078 DRIC-BR-100638 F/G 12/1 NL





MICROCOPY RESOLUTION TEST CHART

ROYAL AIRCRAFT ESTABLISHMENT

Technical Memorandum Mat/Str 1078

Received for printing 9 July 1986

A USER'S GUIDE FOR BIE3D - A BOUNDARY INTEGRAL EQUATION COMPUTER PROGRAM FOR THREE-DIMENSIONAL ELASTOSTATIC ANALYSIS

by

K. W. Man

SUMMARY

This Memorandum describes a three-dimensional numerical stress-analysis computer program which originated at the Department of Mechanical Engineering, Imperial College, University of London. It is based on the Boundary Integral Equation (BIE) method and it is used to carry out stress analysis of three-dimensional components of complex geometry. Stress intensity factors can be derived for the case where the component is cracked. The body is assumed to have linear elastic isotropic properties. The condition of plane strain is assumed at all points along the crack front except where it intersects the free surface, where plane stress assumptions are made. The program is written in standard FORTRAN and it is now operational on the Materials and Structures departmental VAX computer.

Step-by-step instructions have been presented together with examples illustrating the input and output formats.

Validation and accuracy checks of the computer program have been made by solving two bench mark problems where the BIE solutions are compared with known accurate solutions.

LIST OF CONTENTS

		Page
1	INTRODUCTION	3
2	A THREE-DIMENSIONAL BIE COMPUTER PROGRAM FOR AN ELASTIC BODY	4
3	USER'S MANUAL FOR THE BIE3D COMPUTER PROGRAM	5
	3.1 Modelling of problems	5
	3.2 Element subdivision	6
	3.3 Description of the input data	6
	3.4 Mesh-generation program and graphic plotting computer program	9
4	NUMERICAL EVALUATION OF THE STRESS INTENSITY FACTOR	9
	4.1 The displacement method	10
	4.2 The 'Singularity Elements'	11
5	TEST CASES	12
	5.1 Determination of stress *concentration* factors	12
	5.2 Determination of stress intensity factors	13
6	DISCUSSION AND CONCLUSIONS	14
Acknowledgments		15
Appendix A	BIE3D program input data file with descriptions	17
Appendix B	BIE3D program 'condensed' output data file	22
Appendix C	MESH3D program input data file	25
Appendix D	Descriptions of the results of a problem	27
Appendix E	Summary of the Gaussian quadrature coefficients	30
References		31
Illustrations		Figures 1-16
Report documentation page		inside back cover

1 INTRODUCTION

This Memorandum describes a computer program BIE3D, which is used for solving three-dimensional elastostatics problems using the Boundary Integral Equation (BIE) method. This program may be used as a numerical tool for the stress analysis of three-dimensional cracked components in linear elastic isotropic bodies.

The fundamentals of the Boundary Integral Equation method have been described extensively elsewhere (see for instance Banerjee and Butterfield[1]) and will not be repeated here. The boundary surface of the body is divided into small regions called elements; in this program BIE3D, isoparametric quadrilateral elements with eight-nodes are used. The distribution of surface geometry, and the unknown displacements and tractions, are expressed in terms of quadratic shape functions of local coordinates. After the displacements and stresses have been calculated, the basic concepts of linear elastic fracture mechanics are used to determine the stress intensity factors.

The original computer program which was developed in the Mechanical Engineering Department at Imperial College, University of London[2], has been modified and adopted into the Materials and Structures departmental VAX computer. This Memorandum describes its use and lays down the procedures for data preparation and input.

In order to relieve the work load for large problems and eliminate human errors during the process of data preparation, a three-dimensional mesh-generation program has been developed in collaboration with Southampton University[3], and adopted as a data preparation tool; it is presented here as an option to the users. For the purpose of obtaining a three-dimensional representation of the generated mesh a three-dimensional graphics plotting program has been developed at RAE, so that the discretization data can be checked visually in terms of element topology and nodal coordinates before use. The plotting program is written in BASIC and it runs interactively on a Hewlett Packard HP2647F graphics terminal.

The organisation of the BIE3D program as well as the function of each subprogram in BIE3D is briefly described in section 2.

In section 3, the technique of modelling a problem, the element design and subdivisions are described in detail. Also described in section 3 are the guide-lines for the preparation of an input data file. In order to make this preparation as simple as possible, the computer programs for the mesh-generation and graphics plotting have been briefly described in section 3.4 and illustrated with examples in section 5. The full descriptions of these utility programs is described elsewhere[3].

The technique of determining the stress intensity factors is contained in section 4, using a centre-cracked specimen as an example. The BIE results are compared with the values obtained by Newman[4] using the boundary collocation method. In order to estimate the accuracy of the BIE3D computer program the stress concentration factors for a flat plate with a central normal hole have been calculated for comparison with known accurate solutions.

2 A THREE-DIMENSIONAL BIE COMPUTER PROGRAM FOR AN ELASTIC BODY

BIE3D is a computer program for the analysis of elastic behaviour of a three-dimensional body using the method of Boundary Integral Equations. It is written in standard FORTRAN. The body is assumed to have isotropic properties. In order to use this computer program, the surface of the three-dimensional body must be discretized and divided into a small number of eight-node quadrilateral elements. Both the geometry and the variations of the boundary displacements and tractions over these elements are approximately represented by quadratic shape functions.

With these approximations, the general integral equation can be written as a set of simultaneous, linear, algebraic equations which are solved subject to the boundary conditions (either displacement or stress) of the problem. Displacements and tractions of the surface nodal points are produced, from which the stresses at the surface nodes and the displacements and the stresses at any required internal points can be calculated. Detailed descriptions of the BIE3D method and the numerical formulation used in the analysis may be found in Ref 2.

The computer program BIE3D has been written in standard FORTRAN language and it is divided into a main program and a total of 24 subprograms. The order of the subprograms are listed below together with a brief description of each function:

(1) FRMTRX- Forms the matrix and the second member by BIE method (*ie* right-hand side of BIE equations).

(2) MESH- Reads a mesh of eight-node quadrilateral elements.

(3) MESHOUT- Writes out the mesh data.

(4) ARRAY- Puts element node numbers and nodal coordinates in arrays.

(5) TRANS- Transforms cylindrical or spherical system of coordinates to cartesian system.

(6) BCS- Input, process and output of the boundary conditions.

(7) SHAPS- Calculates shape functions and their derivatives.

(8) LINEAR- Calculates linear shape functions and their derivatives.

(9) JACOBI- Evaluates the Jacobian and the components of the unit vector normal.

(10) TRIANG- Calculates the Jacobian for the case where the first argument of the kernels is a node of the element.

(11) KERNEL- Calculates kernels U and T .

(12) INTGR- Carries out the numerical integration of kernel products to form the matrix and second member where the first argument is not a node of the element.

(13) ITSS- Carries out the integration of the kernel products where the first argument is a node of the element.

(14) FMATR- Applies the boundary conditions to form the matrix and the second member.

(15) SOLVE- Solves the linear equations by the BIE formulation. This version treats one load case only.

(16) OUTPUT- Outputs the nodal displacements and the tractions.

(17) STRES- Calculates stresses at surface nodes.

(18) GTRANS- Transforms stresses in the local system of coordinates to the global system.

(19) PSTRES- Calculates the principal stresses at the nodes.

(20) INTPT- Reads the coordinates of the internal points.

(21) STRAIN- Calculates the strain energy of the elastic body.

(22) FORCE- Calculates the equivalent nodal forces of the prescribed uniform stresses.

(23) INTNL- Calculates and outputs the displacements and stresses at internal points.

(24) INTKER- Calculates the kernels for internal points.

3 USER'S MANUAL FOR THE BIE3D COMPUTER PROGRAM

This section describes the operating instructions for the BIE3D program. Descriptions of the results produced for a successfully executed problem are described in Appendix D.

3.1 Modelling of problems

For all BIE methods modelling of problems always starts by discretizing the outside surface which is known as the boundary contour of the domain. The boundary contour of a three-dimensional body is represented by surface elements. The type of element used in the program at present is restricted to an eight-node isoparametric quadrilateral surface element, as shown in Fig 1. The parametric representation of the elements enable curvatures to be modelled where the geometry of the surface is defined in terms of quadratic shape functions of the intrinsic coordinates of the element.

The modelling technique is usually simple, but a decision must be made on how many surface elements are needed to represent any particular problem to the desired accuracy. This is normally dictated by the complexity of the structural shape to be modelled. The accuracy of the results tends to improve as the number of elements used in the modelling of the problem increases, but the computing time and cost increases too. Therefore the user must strike a balance between these factors. Although the modelling of a problem is relatively simple and straightforward some rules must be observed in order to preserve the compatibility of the element nodes between elements.

There are only a few rules which need to be observed when modelling the surface of a three-dimensional body and these are described below:

(a) compatibility of element nodes for all the elements must be preserved;

(b) a numbering convention must be observed when numbering the surface elements. The eight-node quadrilateral element must be defined by a clockwise traverse around the element as viewed on the outside surface;

(c) the eight-node quadrilateral element must have four sides; although the sides need not to be equal in length, they need to be continuous and smooth so that they can be fitted accurately with the quadratic shape function.

3.2 Element subdivision

Any eight-node quadrilateral element may be subdivided into a number of smaller elements. For example, a large square element can be split into two halves by drawing a line through the middle as shown in Fig 2, but in doing so we need to introduce a total of five extra nodal points in order to restore the compatibility of two eight-node elements as shown in Fig 3. Further nodes may have to be introduced in adjacent elements. For example, in the modelling of a three-dimensional body, such as the cube in Fig 4a, the subdivision of one of the elements on one side of the cube causes incompatibility of nodal points with the elements adjacent to it. Further additional nodal points, in all a total of 12 as shown in Fig 4b, must be introduced in order to restore the compatibility of nodes. Generally, element subdivision or a refined mesh will improve the accuracy of the results but the process of element subdivision also rapidly increases the number of elements for a three-dimensional body. The extra elements created increase the amount of input data and the computational time required. However careful mesh design and the use of arbitrarily shaped eight-node elements can improve the modelling efficiency. Further improvement in the formulations of the shape function would allow triangular elements to be used and would provide even greater efficiency in terms of mesh design; a special algorithm is needed for this and it is being developed jointly with Southampton University.

3.3 Description of the input data

For each problem, the program reads in a number of sets of data in the order shown below, the physical units for which must be a self-consistent set. Apart from the TITLE and the specification of the coordinate system, all input data may be entered in the form of free-format. The best way to see how this works is to follow an example which may be found in section 5.1 and the corresponding input data file is listed in Appendix A with descriptions. It should be noted that not all input data defined below may be present for a given problem.

A. Problem title

** Enter problem title; it can be 80 characters long.

B. Quadrature formula data

** Enter the order of Gaussian quadrature formula used for the numerical integration over quadrilateral elements and the corresponding Gaussian abscissae and coefficients. (Note: The Gaussian abscissae and its coefficients are used in the Gaussian quadrature formulae to evaluate various integrals over elements and cells in BIE. A short summary of these values is listed in Appendix E or a comprehensive set of the coefficients may be found in Ref 10. As a guide line fourth- or sixth-order of Gaussian quadrature is sufficient for most cases. If in doubt, the safest way to get information on the

stability of the results is to make several runs with different orders of Gaussian quadrature.)

C. Material properties

** Enter the Young's modulus and Poisson's ratio of the elastic body.

D. Geometric mesh data

** Enter the total number of surface nodal points, the total number of surface elements, the mesh data output control parameter (*ie* ∅ *or* 1) where,

∅ : mesh data output suppressed
1 : mesh data printed out

and the maximum physical dimension of the domain (distance between the two furthest apart boundary elements); this value need only be estimated to a round figure. (Note: Problems requiring total number of nodal points, NNP<332 and total number of elements, NEL<11∅ can be treated without changing the dimensions of the arrays in the program.)

E. Nodal coordinates input

** Enter the type of system of coordinates used,

example

SYS=CAR - Cartesian coordinate system
SYS=CYL - Cylindrical coordinate system
SYS=SPH - Spherical coordinate system

** Enter the nodal point numbers with the corresponding coordinates in accordance with the following table:

SYS -	X(I)	Y(I)	Z(I)
Cartesian	x	y	z
Cylindrical polar	r	θ	z
Spherical	r	θ	ϕ

where the angles θ and ϕ are the usual coordinate angles, and are measured in degrees. In this Memorandum examples are given using Cartesian coordinates only.

F. Element data

** Enter the element numbers and the element node numbers of the right-node quadrilateral element. (The coordinates corresponding to the node numbers were previously defined in E above.)

G. Number of internal points

** Enter the total number of internal point solutions required; these usually are the points of interest, inside the boundary of the body, where stresses, displacements and tractions are required. If no internal solution is required then *zero* must be

entered instead and skip to H. For non-zero values, the coordinates of each interior point are to be entered. The format of the coordinate system for the internal points must be the same as for the surface points.

H. Boundary conditions data

For data input purpose, the boundary conditions are of three types; displacement constraints, node constraints and applied uniform stresses over certain elements.

** Enter the total number of elements with displacement boundary conditions.
(Note: Maximum number of displacement boundary conditions must be less than 11Ø if the dimensions of the arrays in the program are not to be changed.)

** Enter the total number of point constraints where tractions are zero.
(Note: Point constraints are often used in BIE3D to fix the rigid body motion of a body in a chosen direction as in the example in section 5.2. However, the user should ensure when choosing the points of constraint for this purpose that they will not cause any undesirable effects to the final calculated solutions. The maximum number of point constraints must be less than 3Ø if the dimensions of the arrays in the program are not to be changed.)

** Enter the total number of sets of applied uniform stresses.

If there are one or more elements constrained then for each element involved:

** Enter the constrained element number, the direction in which the element is constrained (1, 2 or 3), where

1 : prescribed displacement components in the X-direction
2 : prescribed displacement components in the Y-direction
3 : prescribed displacement components in the Z-direction

and enter the prescribed displacement components.
(Note: The ordering of the elements is arbitrary.)

If there are one or more node constraints, for each node:

** Enter the constrained nodal point number and enter the direction of constraint (1, 2 or 3), where

1 : prescribed displacement components in the X-direction
2 : prescribed displacement components in the Y-direction
3 : prescribed displacement components in the Z-direction

(Note: The ordering of the nodes is arbitrary.)

If there are one or more sets of uniform applied stresses, for each set:

** Enter the total number of elements subjected to the uniform applied stresses and specify the magnitude of the stress fields in Pxx, Pyy, Pzz, Pxy, Pxz and Pyz.
(Note: A combination of these stress fields may be specified so that the simulated load can be characterized.)

** Enter the total number of elements which are not orthogonal to, but share nodes with, those elements acted on by these stresses.
(Note: The maximum number of elements subjected to the uniform applied stresses must be less than 140 if the dimensions of the arrays in the program are not to be changed.)

** Enter the element number of the elements that are subjected to the above set of uniform loads.

END OF FILE

3.4 Mesh-generation program and graphic plotting computer program

Although the input is much simpler for the boundary element method than for the finite element method, most of this advantage would be lost if the coordinates of each mesh-point and the connectivity of each element had to be input separately. Therefore, in order to make the process of creating a data file as simple as possible, a three-dimensional mesh-generation program has been developed and installed in the Materials and Structures departmental VAX computer. The program not only saves time but also eliminates many of the human errors which may occur during manual preparation of the coordinates and element topologies. The mesh-generation program is called MESH3D. It is based on automatic element subdivision of a few large blocks which are defined as input data; each block is then subdivided into elements according to the subdivisions which are provided as input data. MESH3D will calculate all the nodal coordinates for all the subdivided elements. The nodal point numbering convention (clockwise or anticlockwise), generated for the mesh, will be the same as the input data numbering convention. At present, the meshes calculated by the MESH3D are in Cartesian coordinates only.

After a mesh has been generated by the MESH3D, it can be visually checked by plotting out the mesh on a graphics display screen or on paper. The plotting of the mesh is an essential part of the mesh-generation procedure because the only way to check the topology of the mesh quickly and accurately is by visual inspection of a three-dimensional view of the mesh; the large volume and the three-dimensional complexity of the mesh data often make it virtually impossible to check numerically.

A full report[3] and a comprehensive user's guide for both the mesh-generation program and the graphic plotting program have been prepared at the same time as the present Technical Memorandum. Details of the basis of the mesh-generation procedure, subdivision process and the connectivity of individual blocks are explained with examples. In that report, the operations of the graphic plotting program and data handling are also explained with examples.

4 NUMERICAL EVALUATION OF THE STRESS INTENSITY FACTOR

The behaviour of a fatigue crack in a given material is controlled by the parameter called the stress intensity factor, K, which is a measure of the severity of the stress field at the crack tip. The stress intensity factor K_I under Mode I conditions is a function of the loading and the geometry of the crack and the body. The numerical evaluation of crack front stress intensity factors using the BIE method involves two phases:

(a) carrying out a stress analysis of a cracked body using a boundary element mesh, and

(b) development of techniques for the determination of the stress intensity factors after all the displacements and tractions are calculated for all the elements.

It should be noted that in the present program (BIE3D) only those crack problems symmetric about the plane of the crack and under Mode I loading can be analysed; advantage is taken of the symmetry about the crack plane, thus reducing the physical problem size to be analysed. In the modelling of the physical problem, the crack plane is represented as a boundary with appropriate boundary conditions. The crack face is normally free of constraints, but stress constraints may be applied if required. Zero normal displacements are prescribed on the plane ahead of the crack.

Various techniques for determining K_I from the boundary integral results can be used but in the present study only the 'displacement method' will be used; it is explained in the following section.

4.1 The displacement method

For a general three-dimensional crack, the stress intensity factor will vary with the position along the crack front. For an infinite three-dimensional body the relationship between K and displacement in a plane perpendicular to the crack front is the same as in the two-dimensional plane strain case. It is therefore assumed that the relationship between K and displacement in the plane perpendicular to the crack front is given by the two-dimensional plane strain equations if the plane is within the body. For a surface plane it is assumed that the relationship is given by the plane stress equations. These procedures have been discussed by Cruse[8].

The two-dimensional displacement field $[u_1, u_2]$, as shown in Fig 5, at a distance r from the crack tip under Mode I condition is defined as[9]:

$$\begin{bmatrix} u_1 \\ \\ u_2 \end{bmatrix} = \frac{K_I}{4\mu} \sqrt{\frac{2r}{\pi}} \begin{bmatrix} \cos\frac{\theta}{2}\left[\kappa - 1 + 2\sin^2\frac{\theta}{2}\right] \\ \\ \sin\frac{\theta}{2}\left[\kappa + 1 - 2\cos^2\frac{\theta}{2}\right] \end{bmatrix} + \ldots, \tag{1}$$

where suffix 1 and 2 refer to the direction x and y respectively, μ is the shear modulus, and

$$\kappa = \left.\begin{matrix} (3 - 4\nu) & \text{for plane strain} \\ \\ \dfrac{(3 - \nu)}{(1 + \nu)} & \text{for plane stress} \end{matrix}\right\}, \tag{2}$$

where ν is Poisson's ratio. For points along a radial line normal the crack front and in the plane of the crack, the displacement normal to the crack face (*ie* $\theta = \pi$ in Fig 5) is:

$$u_2 = \frac{4K_I}{E'} \sqrt{\frac{r}{2\pi}} \, , \tag{3}$$

where r is the distance from the crack front, and E' is the material stiffness, equal to the Young's modulus, E, for plane stress conditions and equal to $E/(1-\nu^2)$ for plane strain.

If the computed value of u_2 at radial distance r from the crack front is V, the corresponding value of K_I, say K_I^*, is obtained from (3), *ie*

$$K_I^* = \frac{VE'}{4} \sqrt{\frac{2\pi}{r}} \, . \tag{4}$$

The above equation (3) is strictly valid only in the limit when r approaches zero. Thus, values of K_I^* are obtained using (4) from the computed displacements V at a series of nodal points along a radial line on the crack face and extrapolated to $r = \emptyset$ to yield the crack front stress intensity factor K_I. The computed values V in regions very close to the crack front are, however, less reliable and it is therefore necessary to extrapolate from relatively large values of r. This inaccuracy is clearly shown in the worked example described in section 5.2 (Fig 13) where K_I^* is plotted against (r/C), and where C is the crack length. The constant slope portion of the curve is extrapolated to $r = \emptyset$.

The extrapolation method or the displacement method above does not contain any special provision in the numerical formulation for the $r^{\frac{1}{2}}$ dependence of the displacement or the $r^{-\frac{1}{2}}$ singularity in traction at the crack front. Although the boundary integral equation method gives good resolution of the solution parameters, a relatively fine grid around the crack front is still necessary in order to obtain reasonably accurate values of K_I by these extrapolation methods. To overcome this major disadvantage, it is possible for the users to employ the 'quarter point elements' at the crack front in the mesh design. The basic concept underlying the quarter point element is that the requirement for the $r^{\frac{1}{2}}$ dependence of the displacement near the crack front in the 8-node quadrilateral elements can be achieved by placing the mid-side nodes of the elements either side of the crack front at the quarter point instead of the mid-point. The positioning of nodes for quarter point elements is illustrated diagrammatically in Fig 6. It is important to note that this procedure does not require special shape functions in the isoparametric coordinate space and therefore does not require any different computer subprogram. The resolution and accuracy of the solution parameters near the crack tip can be improved further still by the application of the 'Singularity Elements' ahead of the crack front in the numerical formulation of the fracture problem.

4.2 The 'Singularity Elements'

The special 'Singularity Elements', which exhibit the stress (strain) and traction singularities occurring at the crack front, have been used in two-dimensional problems. They require less mesh refinement around the crack-tip region than the extrapolation method, described earlier, to obtain accurate values of the stress intensity factors.

This concept has been introduced from the finite element method and extended to the boundary integral equation method with the *isoparametric formulation*. The basic concept underlying the singularity is that there is a requirement for an $r^{-\frac{1}{2}}$ singularity in tractions which can be achieved by imposing a shape function in the numerical formulation to characterize this $r^{-\frac{1}{2}}$ behaviour near the crack tip. The special formulation needs to be contained in a computer subprogram. However, singularity elements in tractions have been formulated in two dimensions only[7] and no three-dimensional procedures are yet available.

5 TEST CASES

This section contains two worked examples which will be used to illustrate the three-dimensional boundary element program (BIE3D) and how it can be used to calculate stress *concentration* factors and stress intensity factors.

In order to show the method of data preparation and the format layout of an input data file, the input data file of example (1) is shown in Appendix A and the corresponding output data file is shown in Appendix B.

The BIE solutions are compared with known accurate solutions.

5.1 Determination of stress *concentration* factors

A thick flat plate with a circular hole normal to the plate surface, as shown in *Fig 7, is a classical stress concentration* problem and well documented in the engineering field. Many accurate numerical solutions for this specimen are readily available for comparison of results. Therefore it will be used here as a test case for the BIE3D computer program.

For simplicity, only a quarter of the flat plate will be modelled by making full use of the symmetry about the x- and y-axes. The three-dimensional mesh representation of the quarter plate consists of a total of 52 quadrilateral surface elements and a total of 158 nodal points. The numbering system of the mesh for the surface elements and the nodal points is shown in Figs 8 and 9 respectively. The corresponding input data file (FP.DAT) is listed in Appendix A. This input data file may be either typed in by the user following the steps described in section 3.3 or it can be generated using the mesh-generation program MESH3D as briefly mentioned in section 3.4; as a demonstration, the corresponding MESH3D input data file is shown here in Appendix C. By comparing the two files in Appendix A and Appendix C it can be seen that a large saving in terms of data preparation is possible. A user's guide for the preparation of a MESH3D input data file has been prepared in the form of a Technical Memorandum[3].

Finally, the commands required to submit the file (FP.DAT) for the BIE3D execution run are as shown:

```
$ASSIGN FP.DAT    FOR005        (assigned to logical unit 5)
$ASSIGN FP.RES    FOR006        (assigned to logical unit 6)
$RUN BIE3D
```

where FP.RES is the output file of the results, as shown in Appendix B; it contains the

displacements, the tractions and the stresses for all the nodes and all the internal points (if specified) of the quarter flat plate. In this problem, the maximum stresses are tensile in the direction of the applied stress and occur at the sides of the hole on the net section. The stress *concentration* factor can be defined as $k_t = \sigma_y/\sigma$.

In order to demonstrate the variation of stresses through the thickness of the specimen the stresses along one side of the hole are plotted from one side of the surface to the other as shown in Fig 10. The maximum stress *concentration* factor occurs at the mid-plane having a value of 3.26, decreasing to 3.01 at the free surface. Also in Fig 10 are the solutions obtained by Sternberg and Sadowsky[4] using an analytical approach for an infinite plate having the same ratio, T/D, of plate thickness to hole diameter. The corresponding analytical solutions are 3.11 and 2.78 respectively. Because of the effect of the finite width of the plate, the BIE solution should be reduced by approximately 1% for comparison with an infinite plate. However, the BIE solutions remain approximately 3-4% higher than those obtained analytically; this is satisfactory agreement in view of the relatively coarse boundary mesh used.

5.2 Determination of stress intensity factors

The compact tension specimen used for fracture toughness testing (ASTM E399-74) was represented by a Single-Edge-Crack (SEC) specimen, as shown in Fig 11, with crack length, C, width, W, thickness, T, and length, 2H, such that C/W = T/W = 0.5 and H/W = 0.6. Poisson's ratio was taken to be 0.3. Taking advantage of the two-plane symmetry, only one-quarter of the specimen was modelled. The boundary mesh idealization is shown in Fig 12. It consists of 64 surface elements and 194 nodes. The numbering systems of the meshes for the surface elements and the nodal points are shown in Figs 12 and 13 respectively. In order to simulate the compact tension specimen load, a uniform shear stress, σ_{xy} , was applied to the $x = 0$ plane, as shown in Fig 11, to produce a total vertical load in the y-direction; where the uniform shear stress used in this present example was unity.

The stress intensity factors along the crack front were obtained by the extrapolation method with the use of crack face displacements as described in section 4.1. For the present example, the crack face displacements at the free surface plane, mid-plane and symmetry plane were used in order to calculate the stress intensity factors along the crack front. The results are presented in graphical form in Fig 14 where the normalised stress intensity factors $(K_I^*/[F(C)^{\frac{1}{2}}/TW])$ are plotted against (r/C) and the linear portion of the curve is then extrapolated to the crack tip and yields the value of K_I at $r = 0$. It should be noted that the results shown in Fig 14 were obtained with the conventional isoparametric elements. Plane strain conditions were assumed to prevail at all points along the crack front, except where it intersects the free surface and there plane stress assumptions were made.

The variation of the normalised stress intensity factors along the crack front are plotted in Fig 15. Also shown in Fig 15 is the two-dimensional plane strain solution for the actual test specimen geometry obtained by Newman (1974)[5] using the boundary collocation method, which is 13.68. The BIE solution obtained in this case is 13.36, which is about 2% lower at the symmetry plane than Newman's result.

Another problem was analysed using the same specimen. In this case a uniform tensile stress, σ_{YY}, was applied at the y = H plane, as shown in Fig 11. The stress intensity factors were again calculated using the displacement method and their variation across the thickness of the specimen is shown in Fig 16. The stress intensity factors were normalised by $\sigma_0[\pi C]^{\frac{1}{2}}$ in this case, and at the symmetry plane the value obtained was 2.85, while that at the free surface was 2.60. Again, plane strain conditions were assumed to prevail at all nodal points along the crack length, except at the free surface.

No results for the same geometry and loading are available in the literature for comparison. However, a two-dimensional plane strain solution of the normalised K_I of 2.80 have been calculated for a specimen with the same crack length by Bowie and Neal[6] using the conformal mapping technique.

6 DISCUSSION AND CONCLUSIONS

In the present Memorandum a computer program based on the Boundary Integral Equation method has been described. The essential parts of the operating procedure of the program have been demonstrated with examples. It has been shown that with care the computer program can be used as a numerical tool to calculate the stress intensity factors for a cracked body. The method illustrated in the worked examples for the calculation of the stress intensity factors is based on the displacement extrapolation method. Mathematically, the extrapolation method is based on the two-dimensional stress fields in a two-dimensional plane body. Therefore it is not an exact analogy in the three-dimensional cases. However for an infinite three-dimensional body the relationship between K and displacement in a plane perpendicular to the crack front is the same as in the two-dimensional plane strain case. On the other hand, the surface of a three-dimensional body cannot be represented by the two-dimensional plane stress equations. Therefore better methods are needed so that more reliable stress intensity factor solutions can be obtained in the future. Currently, there are two methods being developed jointly in RAE Farnborough and the Mechanical Engineering Department, Southampton University:

(1) Subtraction method[11], and

(2) Bueckner singularity method[12].

A three-dimensional mesh-generation program[3] has been developed as a tool to prepare input data files for the BIE3D computer program. When the mesh-generation program is coupled with the BIE3D program, the user can solve problems with greater efficiency. In order to obtain a visual check of the three-dimensional view of the mesh, a three-dimensional graphic plotting program has also been developed. These computer programs provide a useful part of the software support for the future development of any numerical stress analysis programs to obtain stress intensity factors for cracks in three-dimensional bodies.

Acknowledgments

The author wishes to thank Dr R.T. Fenner and his colleagues of the Mechanical Engineering Department, Imperial College, London, for supplying the original program and for valuable discussions throughout the present work.

Appendix A

BIE3D PROGRAM INPUT DATA FILE WITH DESCRIPTIONS

(The following is a listing of the input data file for the example described in section 5.1.)

```
DATA FILE                                        DESCRIPTIONS
TITLE: FLAT PLATE WITH A CENTRE HOLE        .... Problem title.
6.000000                                    .... Gaussian quadrature.
 0.2386192      0.4679139                   .... Gaussian abscissae and
 0.6612094      0.3607616                        coeficients.
 0.9324695      0.1713245
-0.2386192      0.4679139
-0.6612094      0.3607616
-0.9324695      0.1713245

  1000.000      0.3000000                   .... Young's modulus and Poission's
                                                 ratio.
158  52  1  5                               .... Total number of nodal points, total
                                                 number of elemets, data-output
                                                 parameter and maximum dimension of
                                                 the problem.
CAR                                         .... Cartesian coordinate system.

1   1.00000        0.00000    0.00000       .... Nodal number and coordinates of
2   1.00000        0.00000    0.03500            the nodal points.
3   1.00000        0.00000    0.07000
4   1.00000        0.00000    0.10000
5   1.00000        0.00000    0.13000
6   1.00000        0.00000    0.16500
7   1.00000        0.00000    0.20000
8   1.00000        0.08000    0.00000
9   1.00000        0.08000    0.07000
10   1.00000        0.08000    0.13000
11   1.00000        0.08000    0.20000
12   1.00000        0.16000    0.00000
13   1.00000        0.16000    0.03500
14   1.00000        0.16000    0.07000
15   1.00000        0.16000    0.10000
16   1.00000        0.16000    0.13000
17   1.00000        0.16000    0.16500
18   1.00000        0.16000    0.20000
19   1.00000        0.30000    0.00000
20   1.00000        0.30000    0.07000
21   1.00000        0.30000    0.13000
22   1.00000        0.30000    0.20000
23   1.00000        0.44000    0.00000
24   1.00000        0.44000    0.03500
25   1.00000        0.44000    0.07000
26   1.00000        0.44000    0.10000
27   1.00000        0.44000    0.13000
28   1.00000        0.44000    0.16500
29   1.00000        0.44000    0.20000
```

30	1.00000	0.56000	0.00000
31	1.00000	0.56000	0.07000
32	1.00000	0.56000	0.13000
33	1.00000	0.56000	0.20000
34	1.00000	0.68000	0.00000
35	1.00000	0.68000	0.03500
36	1.00000	0.68000	0.07000
37	1.00000	0.68000	0.10000
38	1.00000	0.68000	0.13000
39	1.00000	0.68000	0.16500
40	1.00000	0.68000	0.20000
41	1.00000	0.74000	0.00000
42	1.00000	0.74000	0.07000
43	1.00000	0.74000	0.13000
44	1.00000	0.74000	0.20000
45	1.00000	0.80000	0.00000
46	1.00000	0.80000	0.03500
47	1.00000	0.80000	0.07000
48	1.00000	0.80000	0.10000
49	1.00000	0.80000	0.13000
50	1.00000	0.80000	0.16500
51	1.00000	0.80000	0.20000
52	0.92350	0.81520	0.20000
53	0.85998	0.69292	0.20000
54	0.73293	0.44836	0.20000
55	0.58470	0.16304	0.20000
56	0.50000	0.00000	0.20000
57	0.85860	0.85860	0.20000
58	0.79421	0.79421	0.20000
59	0.72981	0.72981	0.20000
60	0.60102	0.60102	0.20000
61	0.47223	0.47223	0.20000
62	0.32198	0.32198	0.20000
63	0.17172	0.17172	0.20000
64	0.08586	0.08586	0.20000
65	0.00000	0.00000	0.20000
66	0.81520	0.92350	0.20000
67	0.69292	0.85998	0.20000
68	0.44836	0.73293	0.20000
69	0.16304	0.58470	0.20000
70	0.00000	0.50000	0.20000
71	0.80000	1.00000	0.20000
72	0.74000	1.00000	0.20000
73	0.68000	1.00000	0.20000
74	0.56000	1.00000	0.20000
75	0.44000	1.00000	0.20000
76	0.30000	1.00000	0.20000
77	0.16000	1.00000	0.20000
78	0.08000	1.00000	0.20000
79	0.00000	1.00000	0.20000
80	0.80000	1.00000	0.16500
81	0.68000	1.00000	0.16500
82	0.44000	1.00000	0.16500
83	0.16000	1.00000	0.16500
84	0.00000	1.00000	0.16500
85	0.80000	1.00000	0.13000

86	0.74000	1.00000	0.13000
87	0.68000	1.00000	0.13000
88	0.56000	1.00000	0.13000
89	0.44000	1.00000	0.13000
90	0.30000	1.00000	0.13000
91	0.16000	1.00000	0.13000
92	0.08000	1.00000	0.13000
93	0.00000	1.00000	0.13000
94	0.80000	1.00000	0.10000
95	0.68000	1.00000	0.10000
96	0.44000	1.00000	0.10000
97	0.16000	1.00000	0.10000
98	0.00000	1.00000	0.10000
99	0.80000	1.00000	0.07000
100	0.74000	1.00000	0.07000
101	0.68000	1.00000	0.07000
102	0.56000	1.00000	0.07000
103	0.44000	1.00000	0.07000
104	0.30000	1.00000	0.07000
105	0.16000	1.00000	0.07000
106	0.08000	1.00000	0.07000
107	0.00000	1.00000	0.07000
108	0.80000	1.00000	0.03500
109	0.68000	1.00000	0.03500
110	0.44000	1.00000	0.03500
111	0.16000	1.00000	0.03500
112	0.00000	1.00000	0.03500
113	0.80000	1.00000	0.00000
114	0.74000	1.00000	0.00000
115	0.68000	1.00000	0.00000
116	0.56000	1.00000	0.00000
117	0.44000	1.00000	0.00000
118	0.30000	1.00000	0.00000
119	0.16000	1.00000	0.00000
120	0.08000	1.00000	0.00000
121	0.00000	1.00000	0.00000
122	0.92350	0.81520	0.00000
123	0.92350	0.81520	0.07000
124	0.92350	0.81520	0.13000
125	0.85860	0.85860	0.00000
126	0.85860	0.85860	0.03500
127	0.85860	0.85860	0.07000
128	0.85860	0.85860	0.10000
129	0.85860	0.85860	0.13000
130	0.85860	0.85860	0.16500
131	0.81520	0.92350	0.00000
132	0.81520	0.92350	0.07000
133	0.81520	0.92350	0.13000
134	0.79421	0.79421	0.00000
135	0.85998	0.69292	0.00000
136	0.72981	0.72981	0.00000
137	0.60102	0.60102	0.00000
138	0.73293	0.44836	0.00000
139	0.47223	0.47223	0.00000
140	0.32198	0.32198	0.00000
141	0.58470	0.16304	0.00000

142	0.17172	0.17172	0.00000
143	0.08586	0.08586	0.00000
144	0.50000	0.00000	0.00000
145	0.00000	0.00000	0.00000
146	0.69292	0.89998	0.00000
147	0.44836	0.73293	0.00000
148	0.16304	0.58470	0.00000
149	0.00000	0.50000	0.00000
150	0.00000	0.50000	0.07000
151	0.00000	0.50000	0.13000
152	0.00000	0.00000	0.03500
153	0.00000	0.00000	0.07000
154	0.00000	0.00000	0.10000
155	0.00000	0.00000	0.13000
156	0.00000	0.00000	0.16500
157	0.50000	0.00000	0.07000
158	0.50000	0.00000	0.13000

1	1	2	3	9	14	13	12	8
2	3	4	5	10	16	15	14	9
3	5	6	7	11	18	17	16	10
4	12	13	14	20	25	24	23	19
5	14	15	16	21	27	26	25	20
6	16	17	18	22	29	28	27	21
7	23	24	25	31	36	35	34	30
8	25	26	27	32	38	37	36	31
9	27	28	29	33	40	39	38	32
10	34	35	36	42	47	46	45	41
11	36	37	38	43	49	48	47	42
12	38	39	40	44	51	50	49	43
13	51	44	40	53	59	58	57	52
14	40	33	29	54	61	60	59	53
15	29	22	18	55	63	62	61	54
16	18	11	7	56	65	64	63	55
17	57	58	59	67	73	72	71	66
18	59	60	61	68	75	74	73	67
19	61	62	63	69	77	76	75	68
20	63	64	65	70	79	78	77	69
21	71	72	73	81	87	86	85	80
22	73	74	75	82	89	88	87	81
23	75	76	77	83	91	90	89	82
24	77	78	79	84	93	92	91	83
25	85	86	87	95	101	100	99	94
26	87	88	89	96	103	102	101	95
27	89	90	91	97	105	104	103	96
28	91	92	93	98	107	106	105	97
29	99	100	101	109	115	114	113	108
30	101	102	103	110	117	116	115	109
31	103	104	105	111	119	118	117	110
32	105	106	107	112	121	120	119	111
33	45	46	47	123	127	126	125	122
34	47	48	49	124	129	128	127	123
35	49	50	51	52	57	130	129	124
36	125	126	127	132	99	108	113	131
37	127	128	129	133	85	94	99	132
38	129	130	57	66	71	80	85	133
39	45	122	123	134	136	135	34	41

.... Element numbers and the nodal numbers of the eight-node quadrilateral elements.

```
   40     34  135  136  137  139  138   23   30
   41     23  138  139  140  142  141   12   19
   42     12  141  142  143  145  144    1    8
   43    125  131  113  114  115  146  136  134
   44    136  146  115  116  117  147  139  137
   45    139  147  117  118  119  148  142  140
   46    142  148  119  120  121  149  145  143
   47    121  112  107  150  153  152  145  149
   48    107   98   93  151  155  154  153  150
   49     93   84   79   70   65  156  155  151
   50      1  144  145  152  153  157    3    2
   51      3  157  153  154  155  158    5    4
   52      5  158  155  156   65   56    7    6
0                                                 .... Total number of internal
                                                       point solutions required.

24   1   1                                        .... Total number of displace-
                                                       ment constrained elements,
                                                       total number of displace-
                                                       ment constrained nodes and
                                                       total number sets of
                                                       uniform applied stresses.
1 1 0   2 1 0   3 1 0   4 1 0   5 1 0   6 1 0     .... constrainted element
7 1 0   8 1 0   9 1 0  10 1 0  11 1 0  12 1 0          boundary conditions.
21 2 0  22 2 0  23 2 0  24 2 0  25 2 0  26 2 0
27 2 0  28 2 0  29 2 0  30 2 0  31 2 0  32 2 0
98 3                                              .... constrained node boundary
                                                       conditions.
3    0  1  0  0  0  0     0                       .... Total number of elements
                                                       subject to the uniform
                                                       applied stresses, stress
                                                       components; and total
                                                       number of elements which
                                                       are not orthogonal to, but
                                                       share nodes with these
                                                       elements acted on by
                                                       these stresses.
50  51  52                                        .... element-number of the
                                                       elements that are subjected
                                                       to the above set of
                                                       uniform applied stresses.
NO                                                .... end of data.
S
```

Appendix B

BIE3D PROGRAM 'CONDENSED' OUTPUT DATA FILE

(see section 5)

TITLE: FLAT PLATE WITH A CENTRE HOLE

SPECIFIED GAUSS ABSCISSAE AND COEFFICIENTS FOR ELEMENTAL INTEGRATION IN THE LOCAL CO-ORDINATE SYSTEM

0.23861919E+00	0.46791390E+00
0.66120940E+00	0.36076161E+00
0.93246949E+00	0.17132451E+00
-0.23861919E+00	0.46791390E+00
-0.66120940E+00	0.36076161E+00
-0.93246949E+00	0.17132451E+00

YOUNG"S MODULUS = 0.10E+04 POISSON"S RATIO = 0.3000E+00

GEOMETRIC DATA FOR THE MESH

NUMBER OF ELEMENTS = 52

NUMBER OF NODAL POINTS = 158

SYST	NODE	X	Y	Z
CAR	1	0.1000E+01	0.0000E+00	0.0000E+00
CAR	2	0.1000E+01	0.0000E+00	0.3500E-01
.	.	.	.	.
.	.	.	.	.
.	.	.	.	.
.	.	.	.	.
CAR	157	0.5000E+00	0.0000E+00	0.7000E-01
CAR	158	0.5000E+00	0.0000E+00	0.1500E+00

ELEMENT DATA

ELEM	I	J	K	L	M	N	P	Q
1	1	2	3	9	14	13	12	8
2	3	4	5	10	16	15	14	9
4	12	13	14	20	25	24	23	19
.	.	.	.	.	.	.	.	.
.	.	.	.	.	.	.	.	.
.	.	.	.	.	.	.	.	.
.	.	.	.	.	.	.	.	.
51	3	157	153	154	155	158	5	4
52	5	158	155	156	65	56	7	6

STRESSES AND DISPLACEMENTS AT INTERIOR POINTS NOT REQUIRED.

DISPLACEMENT BOUNDARY CONDITIONS·

CONSTRAINED ELEMENTS

ELEM	NCOND	UPRES	ELEM	NCOND	UPRES	ELEM	NCOND	UPRES
1	1	0.000E+00	2	1	0.000E+00	3	1	0.000E+00
4	1	0.000E+00	5	1	0.000E+00	6	1	0.000E+00
7	1	0.000E+00	8	1	0.000E+00	9	1	0.000E+00
10	1	0.000E+00	11	1	0.000E+00	12	1	0.000E+00
21	2	0.000E+00	22	2	0.000E+00	23	2	0.000E+00
24	2	0.000E+00	25	2	0.000E+00	26	2	0.000E+00
27	2	0.000E+00	28	2	0.000E+00	29	2	0.000E+00
30	2	0.000E+00	31	2	0.000E+00	32	2	0.000E+00

```
CONSTRAINED NODES
NODE     NCOND       NODE     NCOND       NODE     NCOND       NODE     NCOND
 98        3

TRACTION BOUNDARY CONDITIONS
 JNEL        PXX          PYY          PZZ          PXY          PXZ          PYZ
NELT
ELEMENTS OVER WHICH THESE STRESSES ACT
  3          0.           1.           0.           0.           0.           0.
  0
  50         51           52

NODE ICOND ELEM TPRES     NODE ICOND ELEM TPRES     NODE ICOND ELEM TPRES
   1   2   50   0.100E+01  144   2   50   0.100E+01  145   2   50   0.100E+01
 152   2   50   0.100E+01  153   2   50   0.100E+01  157   2   50   0.100E+01
   3   2   50   0.100E+01    2   2   50   0.100E+01    3   2   51   0.100E+01
 157   2   51   0.100E+01  153   2   51   0.100E+01  154   2   51   0.100E+01
 155   2   51   0.100E+01  158   2   51   0.100E+01    5   2   51   0.100E+01
   4   2   51   0.100E+01    5   2   52   0.100E+01  158   2   52   0.100E+01
 155   2   52   0.100E+01  156   2   52   0.100E+01   65   2   52   0.100E+01
  56   2   52   0.100E+01    7   2   52   0.100E+01    6   2   52   0.100E+01

CALCULATED VALUES OF TRACTIONS
 NODE        NCOND    TRACTION
    1          1     -0.4016E+00
    2          1     -0.4342E+00
    3          1     -0.3442E+00
    .          .          .
    .          .          .
    .          .          .
    .          .          .
  112          2     -0.8020E+00
  121          2     -0.8782E+00
  120          2     -0.9896E+00

NODAL DISPLACEMENTS
NODE    U          V          W        NODE     U          V          W
   1  .0000E+00 -.1200E-02  .5150E-04   2   .0000E+00 -.1188E-02  .2963E-04
   3  .0000E+00 -.1186E-02  .1386E-04   4   .0000E+00 -.1187E-02  .8085E-08
   5  .0000E+00 -.1186E-02 -.1384E-04   6   .0000E+00 -.1188E-02 -.2961E-04
   .      .          .          .       .       .          .          .
   .      .          .          .       .       .          .          .
   .      .          .          .       .       .          .          .
   .      .          .          .       .       .          .          .
 153  .2121E-03 -.9562E-03  .8377E-05  154  .2095E-03 -.9557E-03  .4465E-08
 155  .2121E-03 -.9562E-03 -.8368E-05  156  .2108E-03 -.9542E-03 -.1721E-04
 157  .5992E-04 -.1056E-02  .8437E-05  158  .5992E-04 -.1056E-02 -.8422E-05

STRESSES IN THE GLOBAL CARTESIAN SYSTEM OF CO-ORDINATES
NODE   SIGXX       SIGYY       SIGZZ       SIGXY       SIGYZ       SIGZX
   1 0.2839E+00  0.1082E+01 -0.2098E+00 -0.3999E-01  0.2113E-01  0.8660E-02
   2 0.3255E+00  0.1045E+01 -0.1267E+00  0.0000E+00  0.7630E-02  0.7920E-02
   .     .           .           .           .           .           .
```

.	.	.	.	.	.	.
.	.	.	.	.	.	.
157	0.1252E+00	0.1000E+01	0.1243E+00	0.0000E+00	0.0000E+00	-0.8805E-02
158	0.1252E+00	0.1000E+01	0.1243E+00	0.0000E+00	0.0000E+00	0.8811E-02

PRINCIPAL STRESSES AT THE NODES

NODE	SIG1	SIG2	SIG3
1	0.1085E+01	-0.2104E+00	0.2821E+00
2	0.1045E+01	-0.1269E+00	0.3256E+00
3	0.1031E+01	-0.1337E-01	0.3033E+00
.	.	.	.
.	.	.	.
.	.	.	.
.	.	.	.
156	0.1035E+01	-0.5865E-01	-0.1989E-01
157	0.1000E+01	0.1159E+00	0.1336E+00
158	0.1000E+01	0.1159E+00	0.1336E+00

STRAIN ENERGY OF THE BODY NOT NEEDED IN ANALYSIS.

Appendix C

MESH3D PROGRAM INPUT DATA FILE

(see section 5.1)

```
TITLE: FLAT PLATE WITH A CENTRE HOLE
6
.23861919        .46791393
.66120940        .36076158
.93246949        .17132449
-.23861919       .46791393
-.66120940       .36076158
-.93246949       .17132449
1000.0 0.3
32 10 8 3
1  1 2 3 4 5 6 7 8
2  5 4 3 9 10 11 12 13
3  12 11 10 14 15 16 17 18
4  17 16 15 19 20 21 22 23
5  7 6 5 13 12 24 25 26
6  25 24 12 18 17 23 22 27
7  7 26 25 28 29 30 1 8
8  25 27 22 21 20 31 29 28
9  20 19 15 14 10 32 29 31
10  1 30 29 32 10 9 3 2
1  1 0 0
2  1 0 .1
3  1 0 .2
4  1 .4 .2
5  1 .8 .2
6  1 .8 .1
7  1 .8 0
8  1 .4 0
9  .5 0 .2
10  0 0 .2
11  .4293 .4293 .2
12  .8586 .8586 .2
13  .9235 .8152 .2
14  0 .5 .2
15  0 1 .2
16  .4 1 .2
17  .8 1 .2
18  .8152 .9235 .2
19  0 1 .1
20  0 1 0
21  .4 1 0
22  .8 1 0
23  .8 1 .1
24  .8586 .8586 .1
25  .8586 .8586 0
26  .9235 .8152 0
27  .8152 .9235 0
28  .4293 .4293 0
```

```
29  0 0 0
30  .5 0 0
31  0 .5 0
32  0 0 .1
1  3  4
3.5  3  3.5
2  3.5  3  1.5
2  4  1
1.5  3  3.5  2
1
3  4  1
1.5  3  3.5  2
1
4  4  3
1.5  3  3.5  2
3.5  3  3.5
5  3  1
3.5  3  3.5
1
6  3  1
3.5  3  3.5
1
7  1  4
1
1.5  3  3.5  2
8  1  4
1
1.5  3  3.5  2
9  3  1
3.5 3 3.5
1
10  1  3
1
3.5  3  3.5
```

Appendix D

DESCRIPTIONS OF THE RESULTS OF A PROBLEM

Full descriptions of an output file of a successfully executed problem are listed below in this section. Each individual output description in an output file will be explained so that the users can follow the output descriptions and check against their own files if required.

Descriptions produced by the computer program BIE3D will be quoted here as <output description>.

The results produced for a successfully executed problem are printed in the following order. The units are those of the input data which must be a consistent set.

Problem title is printed according to the particular problem title supplied as input data.

Gaussian quadrature formula

A heading:

<SPECIFIED GAUSS ABSCISSAE AND COEFFICIENTS FOR ELEMENTAL INTEGRATION IN THE LOCAL COORDINATE SYSTEM>

is printed followed by a headed table of the Gaussian abscissae and coefficients supplied as input data.

Material property data

The Young's modulus and Poisson's ratio of the elastic body are printed following the messages < YOUNG'S MODULUS = > and < POISSON'S RATIO = > respectively.

Mesh data

If the mesh data output control parameter (MOUT) has the values zero, no mesh data are printed. The value of this parameter should be defined, normally by being read in as data, in subprogram MESH. Any non-zero value causes a full set of mesh data to be printed in the following order:

(i) A heading <GEOMETRIC DATA FOR THE MESH>

(ii) A message < NUMBER OF ELEMENTS = > followed by the relevant number.

(iii) A message < NUMBER OF NODAL POINTS = > followed by the relevant number.

(iv) A headed table of node number and global coordinates for each node in turn in numerical order.

(v) A heading <ELEMENT DATA> followed by a headed table of element number and the numbers of its eight nodes.

Boundary conditions

(i) A heading <DISPLACEMENT BOUNDARY CONDITIONS> is printed. On the next line a heading <CONSTRAINED ELEMENTS> is printed, followed by a headed table showing element number, restraint condition type number (1 or 2 or 3) where

1 : prescribed displacement component in the X-direction
2 : prescribed displacement component in the Y-direction
3 : prescribed displacement component in the Z-direction and the components of the prescribed displacement.

(ii) A heading <TRACTION BOUNDARY CONDITIONS> is followed on the next line by one or more headed table(s) showing the total number of elements under each uniform loading, the three stress components and the numbers of the elements over which the stresses act. This is followed by a headed table showing the element number, the node number, condition type number (1 or 2) where

1 : prescribed traction components in the X-direction
2 : prescribed traction components in the Y-direction
3 : prescribed traction components in the Z-direction and the components of the prescribed tractions.

<u>Nodal point tractions and displacements</u>

(i) A heading <CALCULATED VALUES OF TRACTIONS> is printed followed by a headed table showing the node number, condition type number (1 or 2) where

1 : prescribed traction components in the X-direction
2 : prescribed traction components in the Y-direction
3 : prescribed traction components in the Z-direction and the components of the calculated tractions.

(ii) A heading <NODAL DISPLACEMENTS> is printed followed by a headed table showing the node number and the computed displacement components for each node in turn in numerical order.

<u>Shifted nodes</u>

In order to impose the $r^{-1/2}$ singularity at the crack tip, the mid-side nodes of the elements adjacent to the crack tip are shifted to quarter points nearest to the tip; hence the Jacobian of the transformation from global to local coordinate system becomes zero. This cases the following message to be printed.

<JACOBIAN = Ø . AT NODE = NN ELEMENT NO = MM>

with NN and MM replaced by the relevant numbers. The program is not halted. However, the stresses at that node are not calculated since the stresses would be infinite there.

<u>Nodal point stresses</u>

A heading <STRESSES IN THE GLOBAL CARTESIAN SYSTEM OF COORDINATES> is printed followed by a headed table showing the node number and the SIX stress components for each node in numerical order.

<u>Interior points</u>

If any internal solutions are required a heading <INTERNAL POINT COORDINATES> is printed followed by a headed table showing internal point number and the coordinates of the point.

Displacement and stresses at interior points

(i) The first set of the final set of results to be written are the stresses at internal points. A heading <STRESS AT INTERNAL POINTS> is followed by a headed table showing the interior point number and the three stress components for each point.

(ii) Finally, a heading <DISPLACEMENTS AT INTERNAL POINTS> is followed by a headed table showing the interior point number and the displacement components for each point in turn.

Appendix E

SUMMARY OF THE GAUSSIAN QUADRATURE COEFFICIENTS

	Gaussian abscissae	Gaussian coefficients
	0.3399810435	0.6521451548
4	0.8611363115	0.3478548451
	-0.3399810435	0.6521451548
	-0.8611363115	0.3478548451
	0.2386191860	0.4679139345
	0.6612093864	0.3607615730
6	0.9324695142	0.1713244923
	-0.2386191860	0.4679139345
	-0.6612093864	0.3607615730
	-0.9324695142	0.1713244923
	0.1834346424	0.3626837833
	0.5255324099	0.3137066458
	0.7966664774	0.2223810344
8	0.9602898564	0.1012285362
	-0.1834346424	0.3626837833
	-0.5255324099	0.3137066458
	-0.7966664774	0.2223810344
	-0.9602898564	0.1012285362

REFERENCES

No.	Author	Title, etc
1	P.K. Banerjee R. Butterfield	Boundary element methods in engineering science. McGraw-Hill (1981)
2	C.L. Tan	Three-dimensional boundary integral equation stress analysis of cracked components. PhD Thesis, Department of Mechanical Engineering, Imperial College, London (1979)
3	K.W. Man M.H. Aliabadi	Three-dimensional mesh-generation and plotting routines for use with a boundary element program (BIE3D). RAE Technical Memorandum Mat/Str 1077 (1986)
4	E. Sternberg M.A. Sadowsky	Three-dimensional solution for the stress concentration around a circular hole in a plate of arbitary thickness. *J. Appl. Mech.*, <u>16</u>, pp 27-28 (1949)
5	J.C. Newman	Stress analysis of the compact specimen including the effects of pin loading. ASTM STP 560, pp 105-121 (1974)
6	O.L. Bowie D.M. Neal	US Army Materials Research Agency Technical Report, AMRA TR 65-20 (1965)
7	R.N.L. Smith J.C. Mason	PhD Thesis, Royal Military College of Science, Shrivenham, Swindon (1984)
8	T.A. Cruse	Numerical evaluation of elastic stress intensity factors by the boundary integral equation method. In: The surface crack: physical problems and computational solution, ASME, NY (1972)
9	G.R. Irwin	Analysis of stresses and strains near the end of a crack traversing a plate. *J. Appl. Mech.*, <u>24</u>, pp 361-364 (1957)
10	P.C. Hammer A.H. Stroud	Numerical integration over simplexes and cones. *Math. Tables Aids Comp.*, <u>10</u>, pp 130-137 (1956)
11	L.S. Xanthis M.J.M. Bernal C. Atkinson	The treatment of singularities in the calculation of stress intensity factors using the boundary integral equation method. *Computer Methods in Applied Mechanics and Engineering*, <u>26</u>, pp 285-304 (1981)
12	D.J. Cartwright D.P. Rooke	An efficient boundary element model for calculating Green's functions in fracture mechanics. *Int. J. of Fracture*, <u>27</u>, pp R43-R50 (1985)

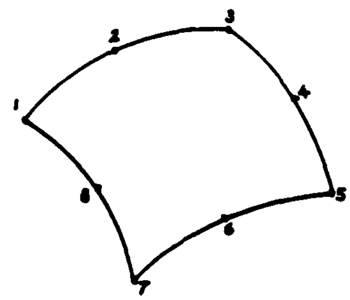


Fig 1 Eight-node quadrilateral element

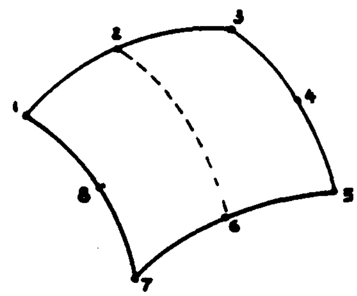


Fig 2 Subdivision of an eight-node quadrilateral element

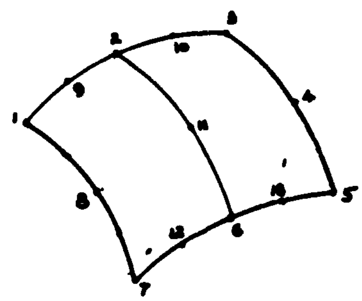


Fig 3 Two equal size eight-node quadrilateral element

Fig 4

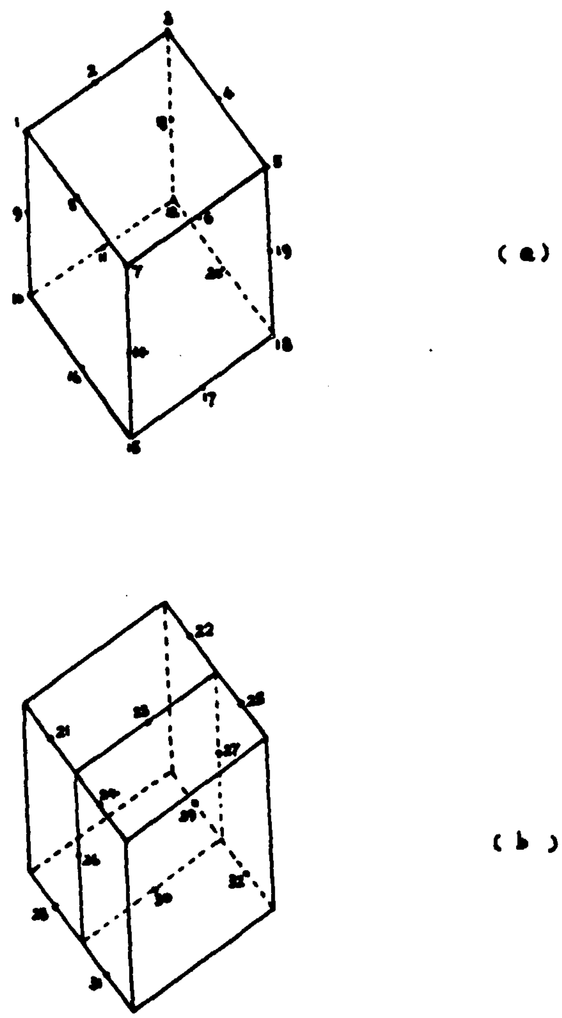


Fig 4 Element subdivision in 3D body

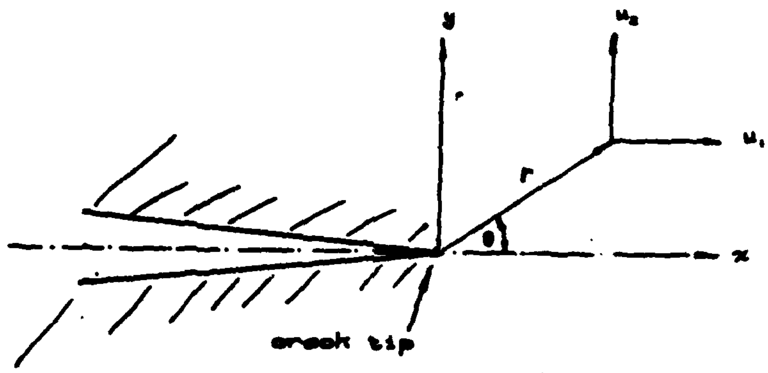


Fig 5 Two-dimensional displacement fields near a crack tip

Fig 6

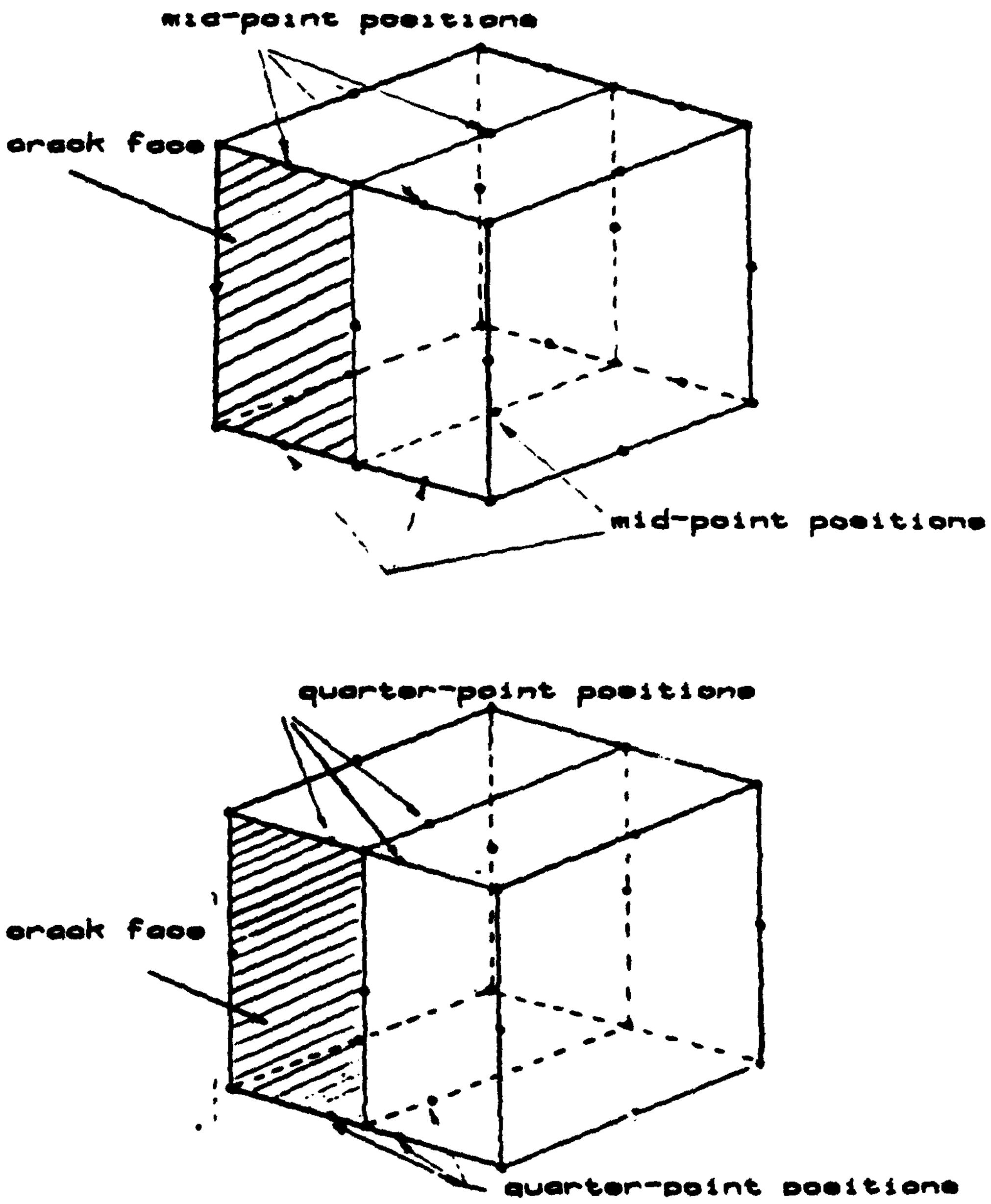


Fig 6 Illustration of quarter-point positions

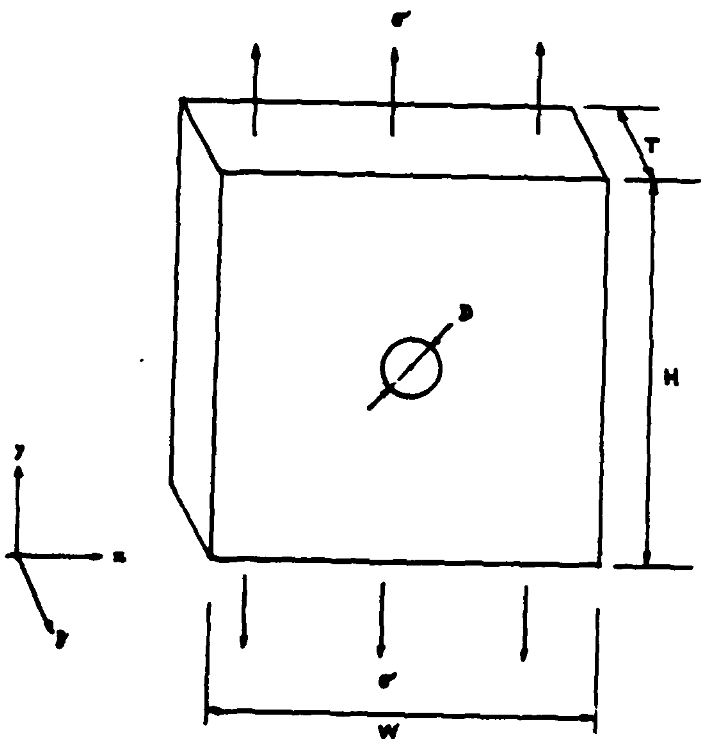


Fig 7 Flat plate with a central normal hole

Fig 8

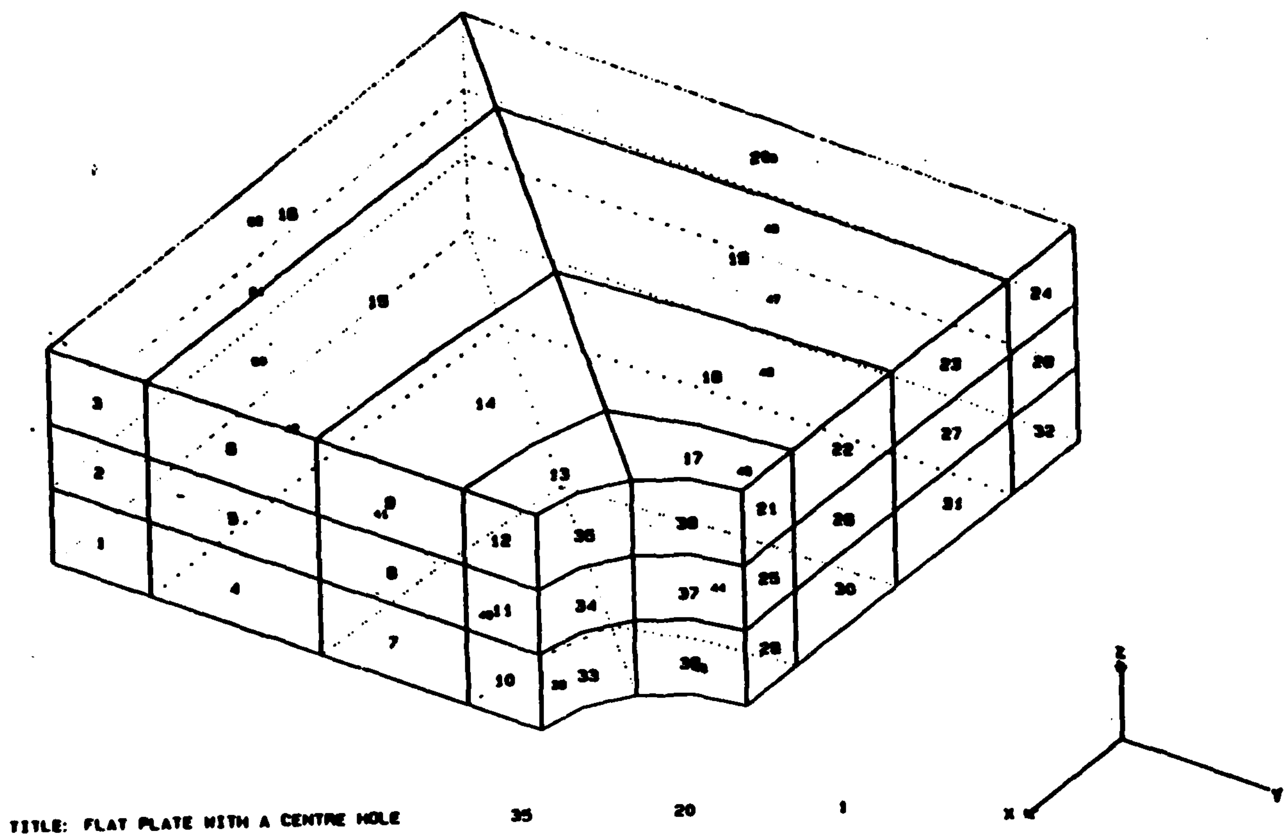


Fig 8 Mesh design and element numbering system of the flat plate

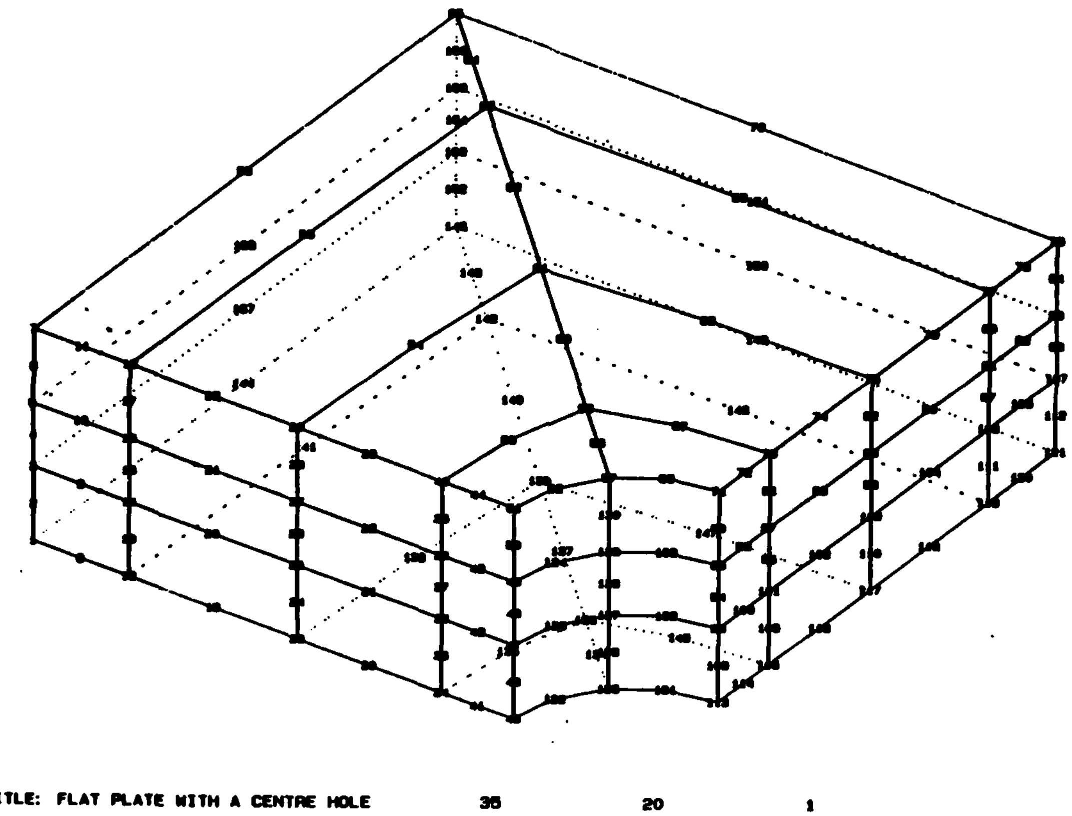


Fig 9 Mesh design and nodal numbering system of the flat plate

Fig 10

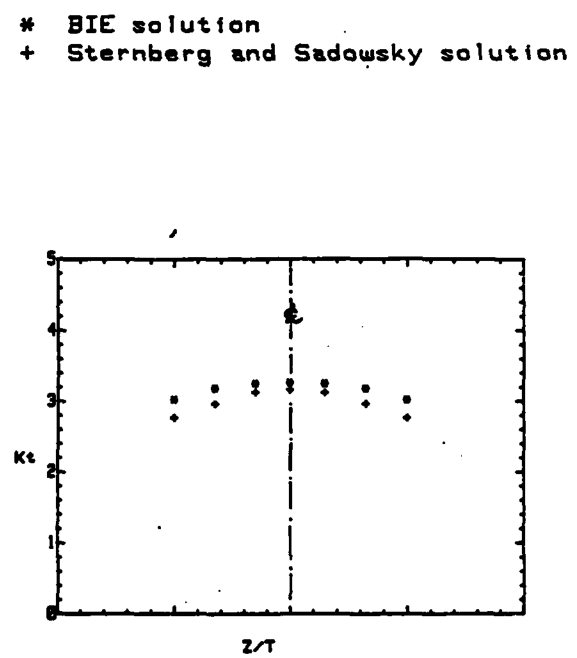


Fig 10 Variation of stress concentration factors at one side of a normal hole

TM MS 1078

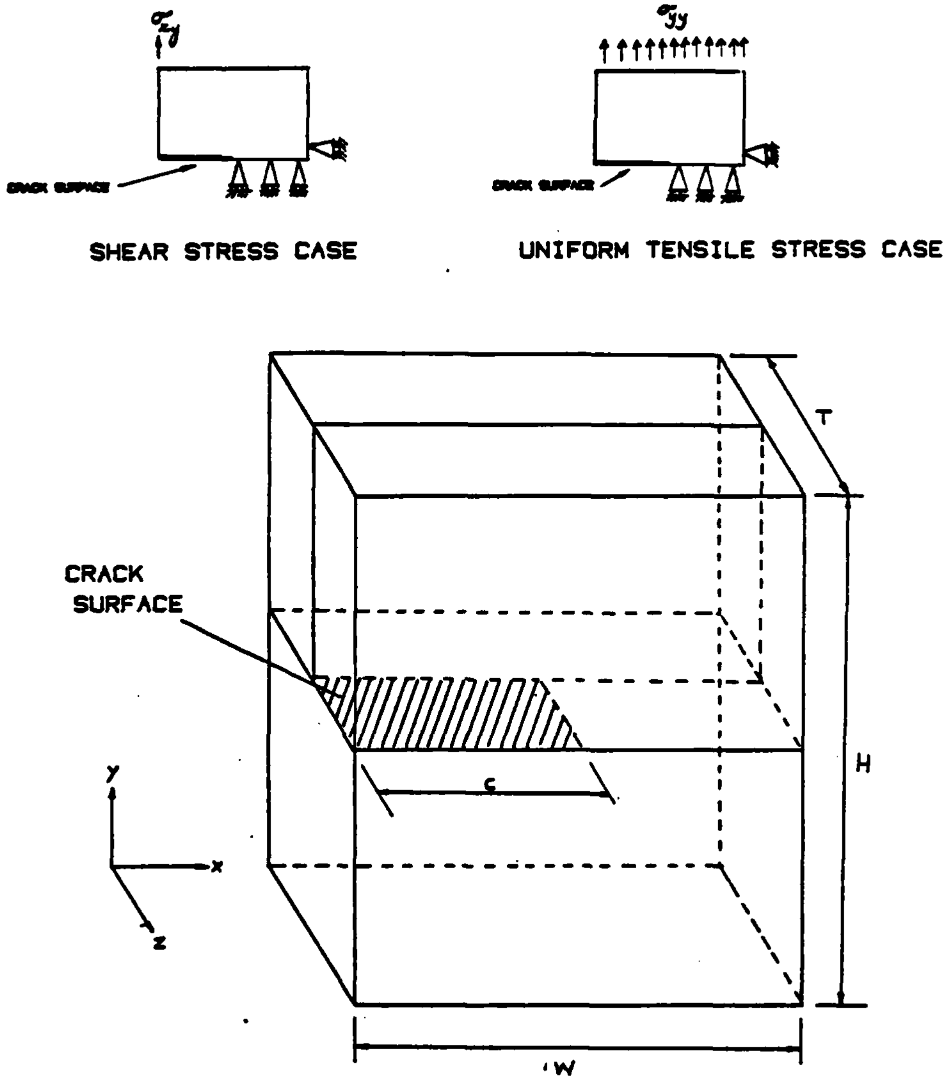


Fig 11 Single edge crack (SEC) specimen

Fig 12

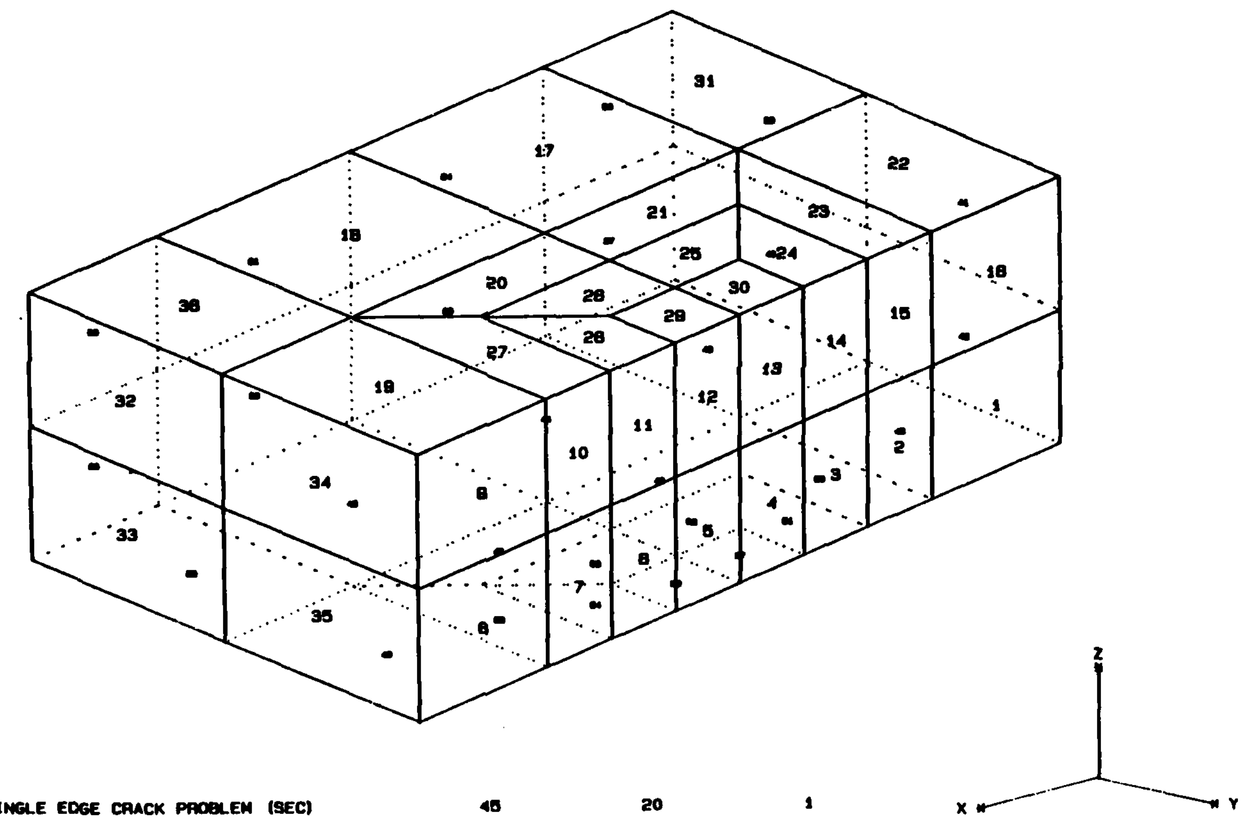
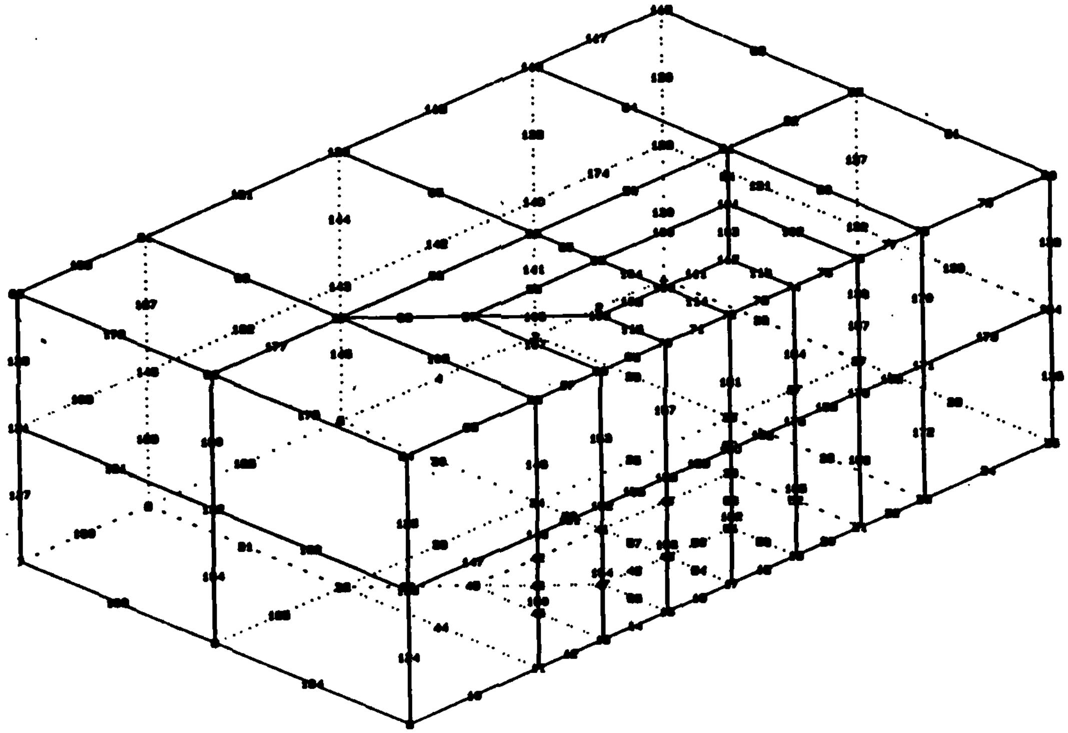


Fig 12 Mesh design and element numbering system of the compact tension specimen



SINGLE EDGE CRACK PROBLEM (SEC)   45   20   1

Fig 13 Mesh design and nodal numbering system of the compact tension specimen

Fig 14

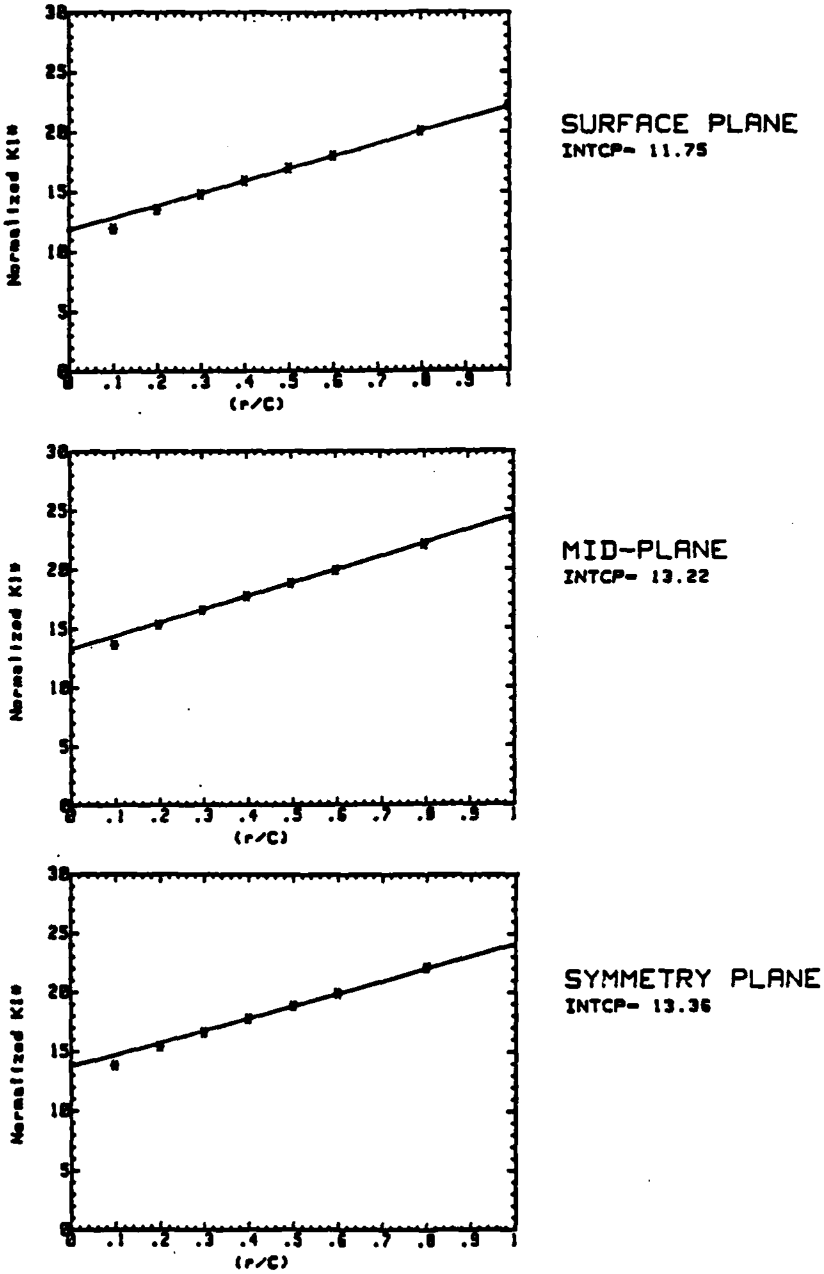


Fig 14 Normalised stress intensity factors ($K_I^*/[F\sqrt{C}/TW]$) against (r/C)

TM MS 1078

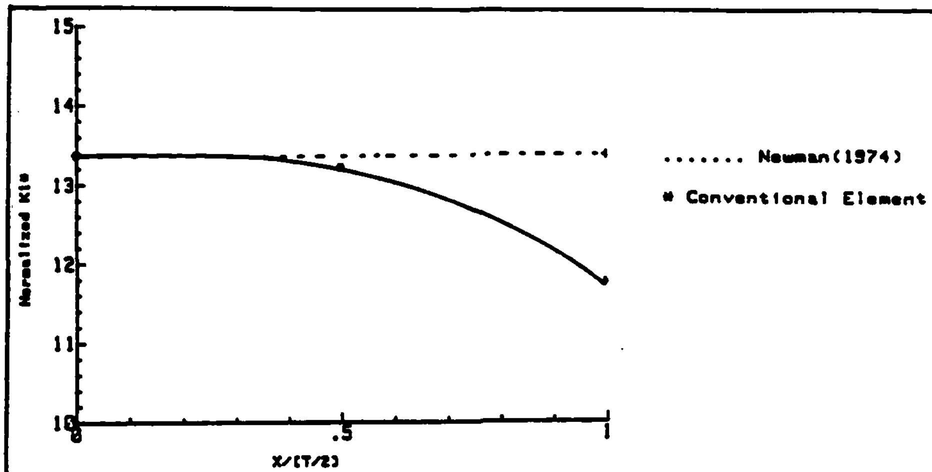


Fig 15 Variation of normalised stress intensity factor ($K_1^*/[F\sqrt{C}/TW]$) with thickness for SEC specimen

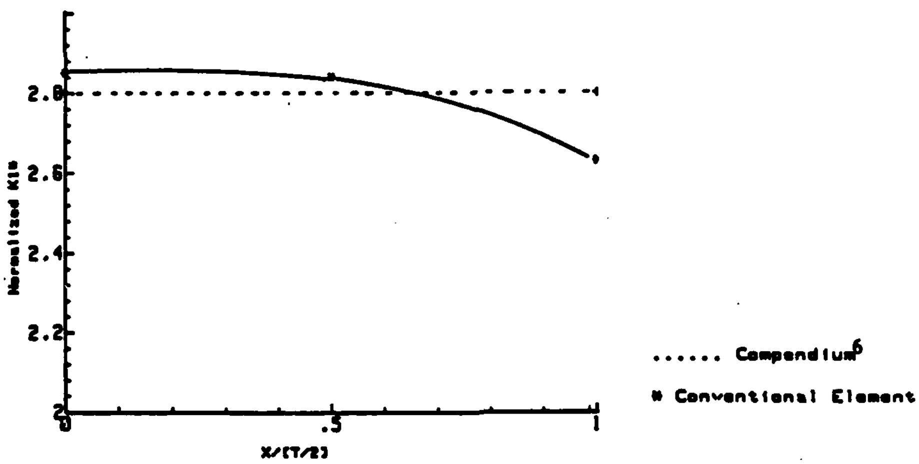


Fig 16 Variation of normalised stress intensity factor ($K_1^*/[\sigma\sqrt{\pi C}]$) with thickness for SEC specimen (uniform tensile load)